ENGINEERING
CREATING A CUSTOM ELECTRONIC COMPONENT LIBRARY USING ORCAD CAPTURE.

Michelle C. Greigg,[a] Istvan Naday*

[a]Miami-Dade Community College, 300 NE 2nd Avenue, Miami FL 33132.

www.mdcc.edu

Abstract

Schematic capture is an integral part of the process of creating an electronic Printed Circuit Board (PCB). It provides the necessary link between schematic design and the creation of a PCB. Typical component libraries comprise of a database of electronic parts that contain most of the information necessary to generate a PCB. However, information about the component's footprint or physical form is omitted. This information is essential to the creation of a PCB and is typically added during the PCB layout and not during the schematic capture. To accelerate the design process, a component library which includes the footprint information was created. This new library created a seamless link between Orcad Capture, the schematic capture program, and Allegro Designer - the PCB layout software. These software programs are key tools in the electronic engineering design process. The new library forms the beginning of a standard set for everyone in the organization to use. The result of this work eliminates time-consuming, inefficient efforts and creates standardization. Building this library required an understanding of the complete PCB design process, learning the intricacies of Orcad Capture, and learning a significant amount about the electronic components entered into the database. This work is an ongoing process because of the enormous number of parts and because new components emerge daily.